Defense Technical Information Center Compilation Part Notice

ADP023087

TITLE: Suppression of Thermal Emission from Exhaust Components Using an Integrated Approach

DISTRIBUTION: Approved for public release, distribution unlimited

This paper is part of the following report:

TITLE: Proceedings of the Ground Target Modeling and Validation Conference [13th] Held in Houghton, MI on 5-8 August 2002

To order the complete compilation report, use: ADA459530

The component part is provided here to allow users access to individually authored sections of proceedings, annals, symposia, etc. However, the component should be considered within the context of the overall compilation report and not as a stand-alone technical report.

The following component part numbers comprise the compilation report:
ADP023075 thru ADP023108

Suppression of Thermal Emission from Exhaust Components Using an Integrated Approach

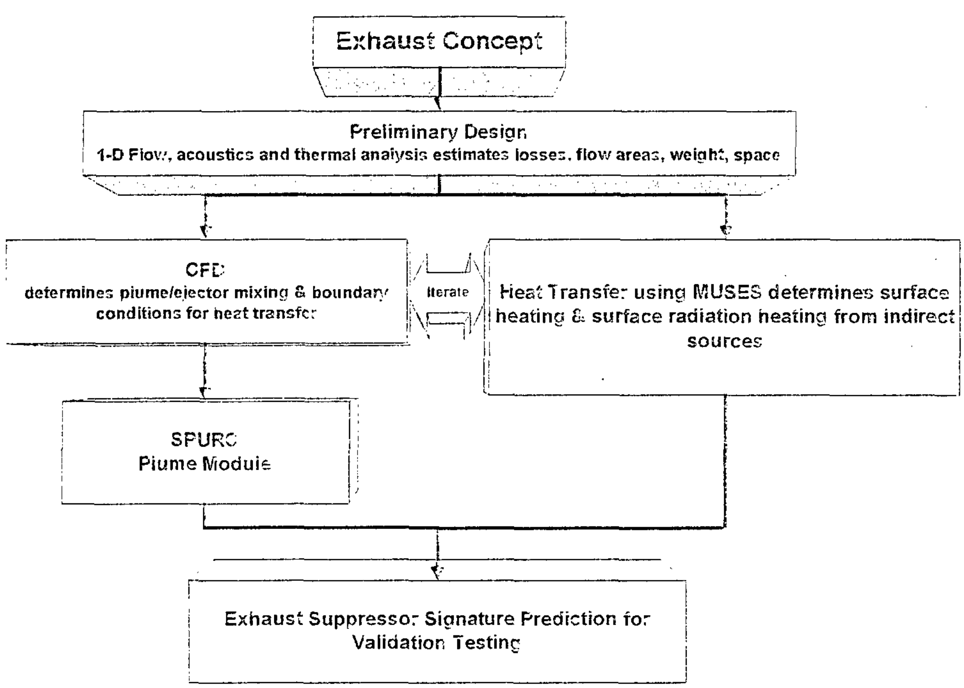
James L. Means, Lance C. Grace
Temeku Technologies Inc.
Lutz, Florida 33548-07222 USA

ABSTRACT

This paper provides the results of an Phase I SBIR to develop an exhaust suppression virtual design model. and exhaust suppression optimization using modeling techniques and hot flow testing. The virtual design model provides an estimate of space required for the exhaust, backpressure to the engine, system weight, gas temperature distributions, and the interaction of the plume with external surfaces both on the vehicle and the ground, and the thermal interaction of the exhaust system with external surfaces through internal convection, conduction, and radiation. A hot flow exhaust test rig has been designed that simulates both exhaust flow and engine bay cooling air flow. This is accomplished with two separate gas producers mounted on flat beds so that the rig is mobile and the sources can be positioned independent of each other.

Virtual Design Model

The virtual design model (VDM) must encompass the disciplines relevant to the performance of an exhaust system. The exhaust system is defined as the hardware necessary to vent the exhaust from the vehicle beginning at the exhaust plane defined by the engine manufacturer and necessary to isolate the exhaust thermally from vehicle structures. The virtual design model must, as a minimum, include an accurate estimate of space required for the exhaust, backpressure to the engine, system weight, gas species distributions, gas temperature distributions, the interaction of the plume with external surfaces both on the vehicle and the ground, and the thermal interaction of the exhaust system with external surfaces through internal convection, conduction, and radiation. Figure 1 depicts the schematic for the virtual design model.



• **Figure 1: Multi-spectral Exhaust System Virtual Design Model**

The modules of the VDM will vary depending on the user and his organization. The VDM should serve as a guide to the use of those modules and provide the input to each module. The VDM must help the exhaust designer interface with CAD

models, structural analysis, ballistic analysis, reliability models, etc. This effort will identify the steps to design and test an exhaust system, sources of data. interfaces with standard design processes, and the degree to which this system is automated.

Computational Fluid Dynamics

CFD analysis of exhaust systems can take weeks to converge and are very difficult to model since the flow regimes include high subsonic compressible internal flow, low velocity wake regions, and regions with high vorticity. To be useful in the design process. the CFD analysis must be responsive to the design process. This can be accomplished by selecting an appropriate CFD code and using it on a subset of the full exhaust system to screen designs, e.g. centerline analysis for high aspect ratio 2-d nozzles.

There are many CFD codes available commercially and in the public domain. The emphasis in this program is to select one of the public domain codes to incorporate in the exhaust design model. Of these codes CFL3D, VULCAN, FUN3D, HEFSS, and WIND were selected as the most viable. CFL3D is a public domain CFD program maintained by NASA Langley Research Center. CFL3D has been used extensively by the U. S. Naval Surface Warfare Center, Carderock Division, and NSWCCD for predicting the flow in exhaust systems and over ship superstructure. All of the commercial codes are available for application on Windows NT. CFL3D is available only for UNIX workstations and supercomputers. However, the FORTRAN code is available and can be hosted on an NT based personal computer.

The future for CFD codes are codes using unstructured grids like FUN3D. NASA representatives feel that good results have been obtained in both wakes and wall regions using the 1998 k-ω turbulence formulation. The flow equations will be stiff and convergence will be difficult. The solution will be strongly dependent on the grid. Results of other CFD studies indicate that the solution will also be very sensitive to the specification of the initialization of the variables in the grid. This agrees with results previously obtained. The emphasis is consolidating existing codes into an advanced CFD code. This is High Energy Flow Solver Synthesis, HEFSS. At the present time there are two codes available and support by the Aerodynamic and Acoustics Methods Branch: VULCAN and CFL3D. Both codes will run on a PC if the PC is running LINUX or another version of UNIX. The Fortran code is available and can be ported onto Windows. VULCAN can compute results with chemical species.

CFL3D has been selected as the CFD code for initial use in the VDM since it is a mature code.

Heat Transfer

The heat transfer analysis must include: Radiation between components; Non-isotropic properties; Adequate bi-directional reflectance (BDRF) model; Interaction with foreground/background; Internal radiation and convection; Compatible with advanced treatments; Secondary Heating due to plume; Run time compatible with design process.

Of the many heat transfer and radiation models available, MUSES is far and away the best. It is already being interfaced with CFD codes to get better local convection coefficients.

Plume Analysis

The plume analysis must include: Multiple exhaust plumes; Non-axisymmetric plumes; Specie distributions for diesel/turboshaft engines; Cross-flow; Interaction with ground plane. The only real alternative for plume signature prediction is the adaptation of SIRRM III using the flow field predicted by the CFD program.

Hot Flow Test Rig

The Phase II SBIR will provide a ground vehicle exhaust hot flow test rig that is capable of testing exhaust systems for ground vehicles with engines (up to 700 hp). This plan will be used to identify the site for the test rig, safety and operational issues, sources of components for the test rig, and serve as a basis for estimating the cost of the test rig.

The criteria for a multi-spectral exhaust system test rig include: Ability to control temperature, pressure, mass flow rate independently ; Isolation of diverted flow from exhaust flow; Simulation of External flow for the real installation; Far enough

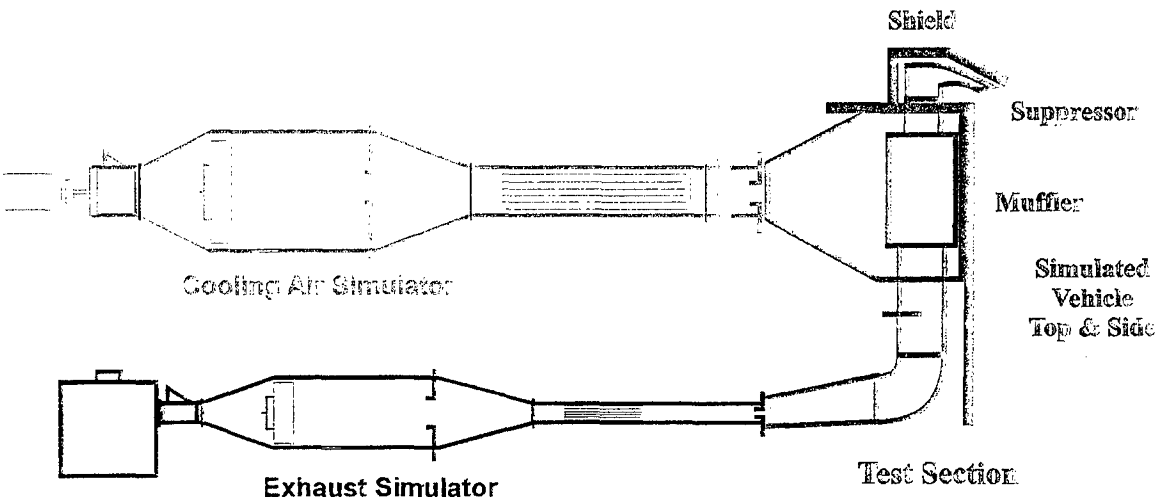
from walls and obstacles to insure that the flow at the test section is not influenced; Compliant with community noise and pollution laws.

The test rig must be able to simulate the full operating range of the exhaust system from idle to full power and must accommodate combinations of engine off-design performance. This is best accomplished when the heat source, the mass flow rate, and the exhaust temperature are controlled independently. This can be accomplished by using a fan with excess flow rate capacity at a backpressure corresponding to the test rig flow loss. Excess mass flow is vented through a waste gate. The waste gate helps obtain the fan maximum pressure by approaching the operating point of the fan from the unstalled side of the fan map. If an engine is used to provide the flow rate and temperature, it must either be the engine for the exhaust system or is capable of providing more than the required mass flow rate and enough gas temperature margins so that the flow can be adjusted to the exhaust system design point.

The optimal exhaust system will function through the full range of flow from idle through max power. Maximum power will correspond to a vehicle operating condition. The optimal test rig will simulate the vehicle installation and the external flow conditions corresponding to the engine power setting. Since most ground vehicles spend a great deal of time at idle, it is also important to be able to simulate cross winds so that secondary heating of vehicle surfaces can be investigated.

Rig Concept

The test rig will have two independent air sources. One air source is required to simulate engine exhaust with enough pressure rise and mass flow rate to simulate engine exhaust at the manifold and the extra pressure drop required by the rig components. Another air source is required to simulate the engine compartment cooling air with enough pressure rise, temperature rise and mass flow rate to simulate the engine compartment pressure drop and rig component pressure drop. Figure 2 depicts this concept.



• **Figure 2: Rig Concept**

The exhaust simulator is designed to provide up to 3,100 standardized cubic feet per minute (SCFM) of air at 60 inches of water pressure at 1,100° Fahrenheit. The engine cooling compartment simulator is designed to provide up to 25,000 SCFM at 12 inches of water pressure at 250° Fahrenheit.

Both systems are very similar. A variable amount of airflow at varying amounts of pressure is input at the left side of the figure. A controllable dump door allows fine-tuning of the air mass and air pressures. A diverging transition section presents the airflow to the combustion section where a propane burner increases the temperature to the desired level. A pitot tube and thermocouple array in the combustion chamber measures conditions that are compared to the measurements from a similar array downstream in the exit tube to provide accurate mass flow rates. A converging transition section smoothly forces the airflow into the exit tube. The air flows through an air straightener before it flows past the second measurement array. The air

exits the right side where extension sections and test item specific transition pieces present the controlled airflows to the test items.

Suppression Techniques

The suppression techniques used for the exhaust system must accommodate suppression in infrared, radar, visual and acoustic spectra. The techniques used in each spectrum must be compatible with the solutions in the other spectra. The Virtual Design Model must be able to analyze and assist in the design the full spectrum of suppression techniques. The Test Rig must be capable of testing and evaluating all of the developmental issues with each of the suppression techniques.

Infrared Suppression Techniques

The techniques used to suppress exhaust signatures are:

- Plume Signature Suppression: Ejector dilution; Fan augmented dilution; Multiple Exhaust plumes with self-absorption and external mixing; Swirl Augmented dilution; Clean exhaust gases
- Hot Part Signature Suppression: Thick Film Cooling; Thin Film Cooling; Fin cooling; Hidden Sacrifice Surface using Coanda Effect; View hiding with insulation; Emissivity Control

Plume Suppression Techniques

Most exhaust systems on diesels and turboshaft engines have exhaust gas temperatures between 850° F and 1000° F. Dilution ratios of 110% to 150% are usually required to meet the desired plume signature. The most direct way to dilute the flow is to use an ejector or a fan to pump cool air into the flow. To insure that the flow is mixed before the exit plane a mixing tube 10 to 20 characteristic dimensions long must be provided in the design. A fan system will require space for either a hydraulic drive, an electric motor, or a mechanical drive from the engine. The power required for the fan system is usually more than the power lost to back pressure for an ejector system. A low loss flow path to ambient air with acceptable radar, visual, and acoustic signatures must be provided for either the fan or ejector to function properly.

An alternative or a complement to the fan or ejector dilution scheme is to use external mixing. External mixing can be promoted to limit the extent of the hot core and provide an area average radiance that is acceptable. There are several ways to promote external mixing. One of the easiest is to break the exhaust flow into multiple jets. This increases the surface area available for mixing, spacing between the jets helps maximize the flow induced for mixing with the jets, and the cool air layers between the jets causes self absorption which reduces the radiance to the seeker. Usually the core temperature of the jet dissipates in 10 to 15 characteristic dimensions from the exhaust plane and nearly completely dissipated in 50-100 characteristic dimensions. Consequently, reducing the characteristic dimension reduces the extent of the hot plume. Swirl also reduces the extent of the plume and enhances external mixing by increasing the path length of the flow and increasing the diffusion of the jet. Combustor designers have long used this principle to maximize mixing in short axial distances.

The primary source of plume radiation is the particulate in the exhaust. Advanced mufflers using catalytic surfaces are available to convert the particulate and CO into CO_2 where there is significant atmospheric absorption.

Hot Part Signature Suppression

Hot part signature suppression can be handled directly by cooling the walls with thick film cooling, thin film cooling or fin cooling to a secondary fluid. Thin film cooling schemes introduce a thin film along the wall and replenish the film before the mixing layer reaches the wall. This introduces a complex structure with many edges to scatter radar energy and little room for radar absorbing material. Thin film cooling is also very susceptible to mal-distribution of flow caused by cross flow resulting in local hot spots. Thick film devices introduce a large flow in a slot that persists for long wall lengths. These devices can be unstable causing transient heating on viewable surfaces. Fin cooling to a secondary fluid attempts to maintain an acceptable wall temperature by increasing the heat transfer rate to such an extent that the difference between wall temperature and gas temperature is maximized.

A second class of hot metal suppressors hides a direct view of hot parts from the threat sensor. This can be used in combination with a thick film cooling/dilution system while using a hidden surface to stabilize the flow and the Coanda effect to turn the flow in a desired direction at the exit plane. When view hiding is used, care must be given to using the

appropriate insulation to prevent conduction or radiation heating of viewable surfaces. Cool surfaces may reflect radiation from hot parts to the threat sensor. If the emissivity of that surface is 1, there will be no reflection. But parts with an emissivity below 1 can have unacceptable reflection of internal hot parts.

Validation Concept

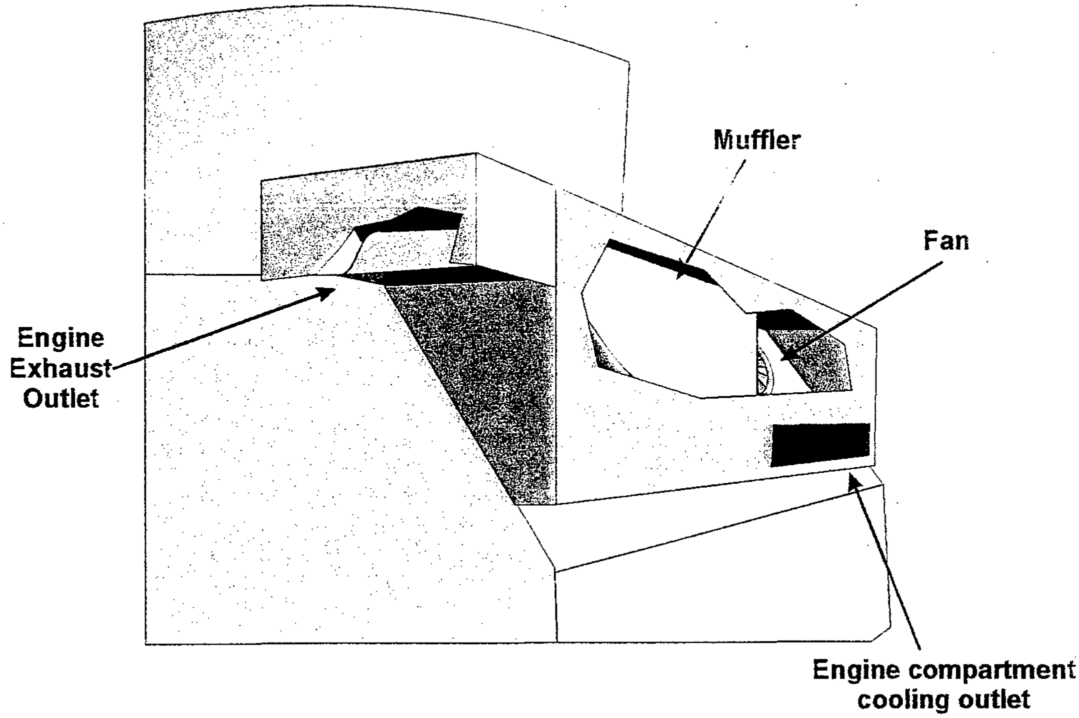
The design to be used as a test case should incorporate:

- Simple Exhaust using Coanda Effect to direct flow away from the vehicle;
- Segmentation into multiple jets to test self absorption;
- Vortex generators to test external mixing;
- Thermal Isolation of Outside surfaces with insulation:
- Redesign of muffler to include removal of particulate while suppressing noise;
- Induced flow to aid in internal heat rejection;
- Structured surface and RAM for other spectra;

The validation design will have three variants to test different aspects of the VDM and the viability of the Hot Flow Test Rig.

The multi-spectral exhaust system validation concept, Figure 3, is composed of an external sponson constructed over the existing facet on a ground vehicle. The engine cooling exhaust is relocated to the sponson where it exhausts out of a LO exit. Cooling air is induced from the inboard interface with the hull and exhausted above the upper lip of the exhaust to prevent the lip of the duct from heating. Insulation with a low emissivity and radar absorbing material will be applied to the interior fan transition duct. The cavity between the transition duct and the sponson will need a radar absorbing material to make any radar return from the cavity negligible. If possible, it is desirable to use a radar absorbing material that acts as insulation and has good acoustic suppression attributes. Cooling air for the lower surface, which is visible to the threat, will be drawn through a slot below the sponson at the interface with the hull. These surfaces will require shaping and radar absorbing material in addition to a structured surface to prevent any reflections from the hot parts above. The transition duct will be attached to the engine exhaust duct before the muffler.

Temeku has developed a concept to insure adequate jet penetration velocity in the engine exhaust at low power settings, the variable flow control. The variable flow control is a blow in door on the exhaust duct. The position of the blow-in door is determined by the relative static pressure between the engine exhaust flow and the fan exhaust flow. The door will weather vane to a position where the two static pressures are equal. When the fan exhaust cannot balance the static pressure of the engine exhaust flow, the door closes and seals. At low engine exhaust flow rates typical of idle, the door weather vanes to equilibrium position and allows fan exhaust air to dilute the flow. Since the cooling fan usually does not have a great deal of back pressure, a low flow loss path must be placed above the muffler to a position just before the exhaust exit so that the fan will be able to pump dilution air. The cooling air exhaust areas will be chosen to permit adequate fan flow to provide the required cooling for the engine while maintaining a high enough pressure in the transition duct to operate the blow-in door.



• **Figure 3: Multi-Spectral Exhaust System Validation Concept**

The first variant will be a simple duct and advanced muffler with advanced insulation. The duct will turn the flow to the horizontal with ejector mixing near the exit. Radar absorbing material will be simulated and the internal geometry will be designed to result in multiple bounces in the radar absorber. A source of ambient air will be provided to the rear surface. A second variant will incorporate the advanced muffler and advanced insulation, but divide the exhaust duct into three separate jets to test the effect of self-absorption and plume interactions. The third variant will incorporate vortex generators in each of the three ducts to test the impact of exit vorticity and the ability of the VDM to predict the flow field.

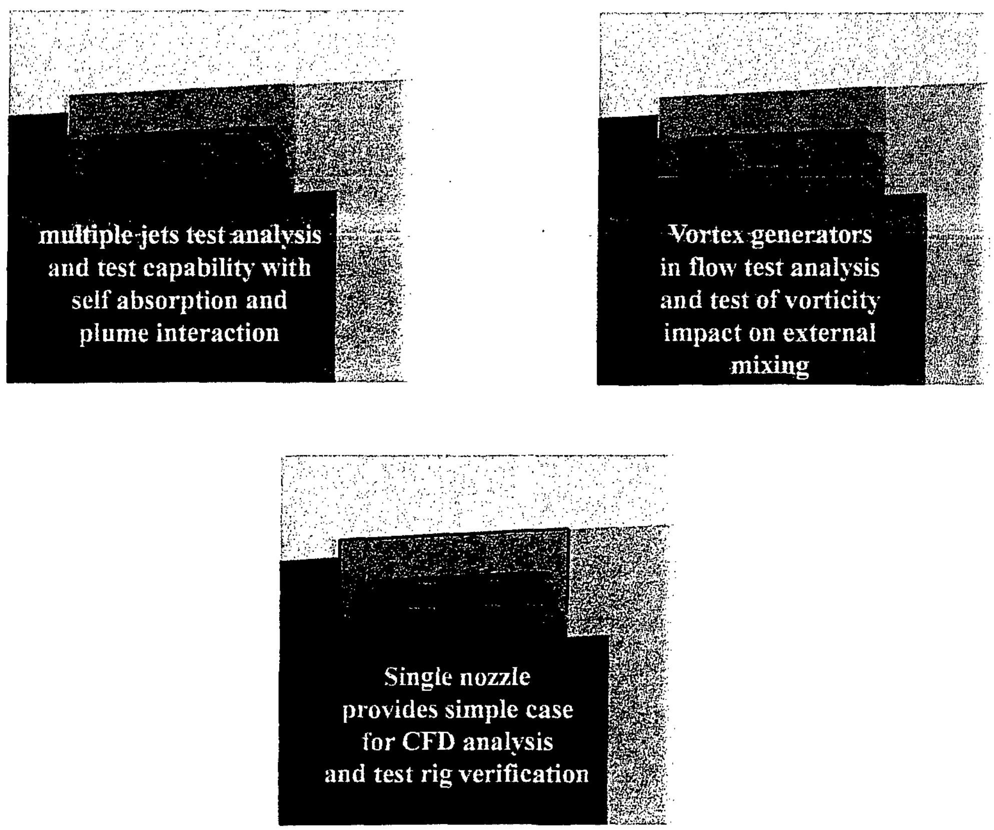
Three exhaust configurations will be tested, Figure 4, during the Phase II effort. The end of the vehicle and the aft portion of the simulated sponson will be simulated on the test section. The simple exhaust without the blow-in door was measured during the Phase I option. During phase II, the complete exhaust system will be simulated with the simple exhaust with the variable flow control, the three nozzle exhaust and the three nozzle exhaust with vortex generators.

The simple nozzle with the blow in door will test the analytical capability to predict the mixing and temperature profiles at the exit. It will test the ability of the test rig to control both flow systems and isolate the control of the two flow systems when they are coupled.

The three nozzle exit will test the analytical capability to predict and optimize induced cold air layers between multiple nozzles and establish the impact of self-absorption on infrared signature. This design will test the ability of the test rig to simulate significant secondary flow paths with isolation from the test rig inlet and waste gate air. It will also test the ability to the rig to measure and document external mixing.

The third concept adds vortex generators to the three-nozzle exit. This will test the analytical capability of predicting external mixing with significant vorticity. Since this exhaust concept spreads radially due to the centrifugal forces on the

exhaust jet, it will stress the test rig's ability to simulate the ground plane with the appropriate thermal characteristics for ground spot analysis.



• **Figure 4: Exhaust duct configurations for analysis, test and validation.**

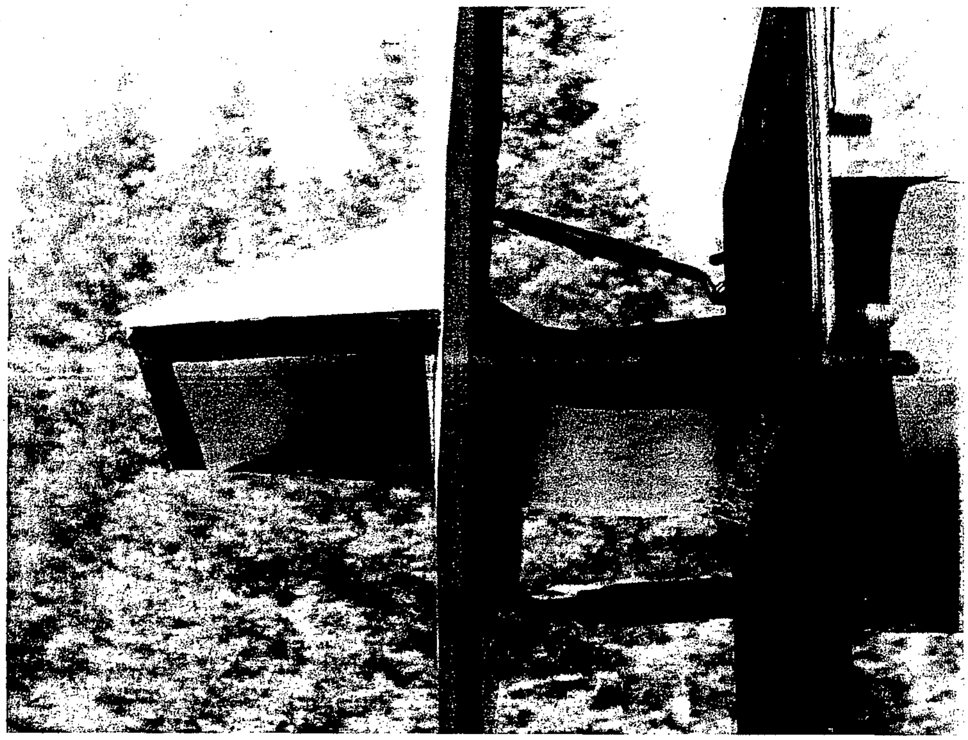
Validation Concept Preliminary Design

Temeku selected a generic design for a typical infrared suppressor for the test nozzle. It was covered by a generic shaped hood. Factors considered for the shapes of both pieces have been previously discussed. The nozzle was fabricated from 1/16 inch thick steel. The hood and simulated vehicle surface were built of wood.

Test Item Description

Temeku selected a generic design for a typical infrared suppressor for the test nozzles. The nozzles were covered by generic shaped hoods. Three separate nozzle configurations were built and tested. Each nozzle had the same side dimensions. In addition, each nozzle had the same cross-sectional throat area of 13 ½ square inches.

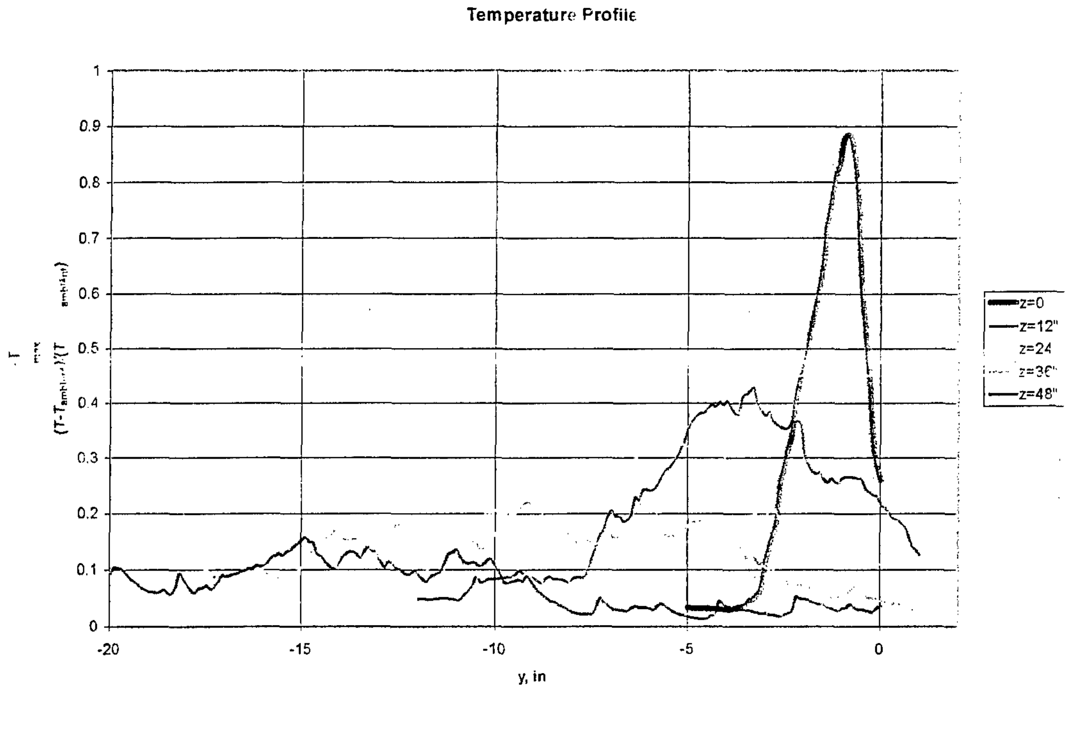
The first nozzle was a single nozzle 9 inches wide. The second nozzle consisted of three 3 inch wide nozzles separated by 3 inch spaces. The third nozzle was the same as the second nozzle but with 4 vortex generators positioned at the center of each wall at the base of the nozzle. The vortex generators in the center nozzle caused clockwise airflow as viewed from outside while the vortex generators in the outside nozzles caused counter-clockwise airflow.



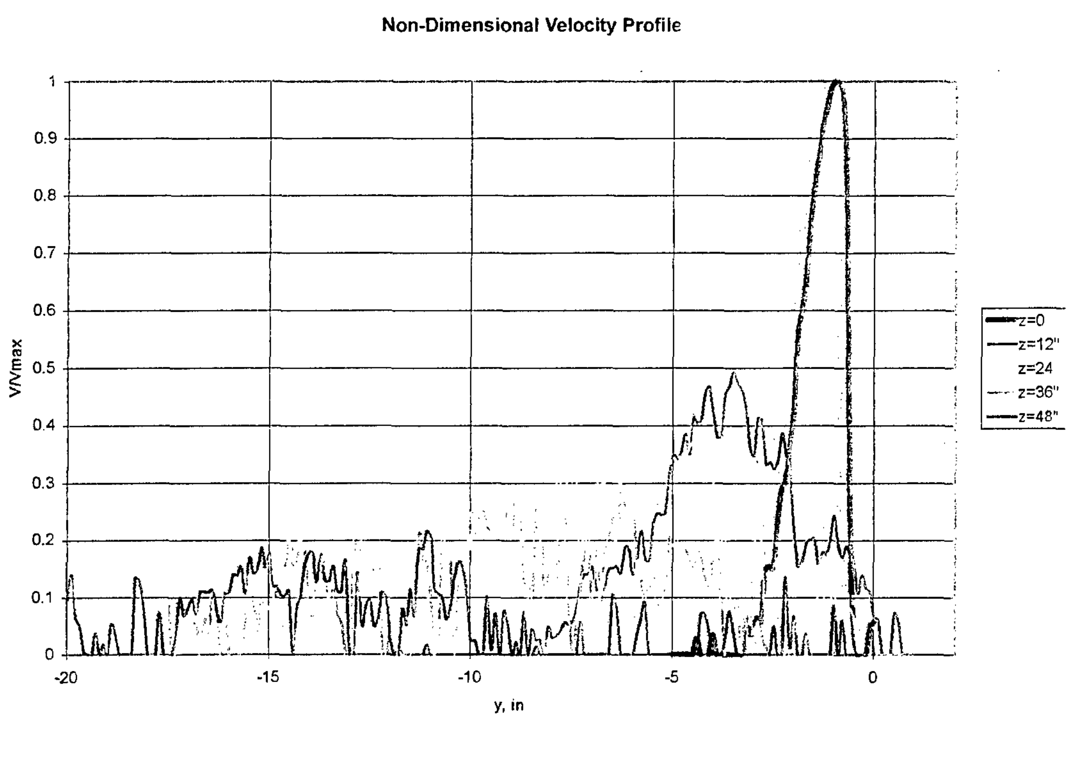
• **Figure 5: Single Nozzle w/ Hood from Side**

Phase I Test Results

All nozzles were tested at a mass flow rate of 0.9 lb/sec at 680° F. The single nozzle results along the center line indicated that the core flow had already mixed at the nozzle exit and the hot flow was essentially dissipated 36" (12 characteristic dimensions) from the exit.

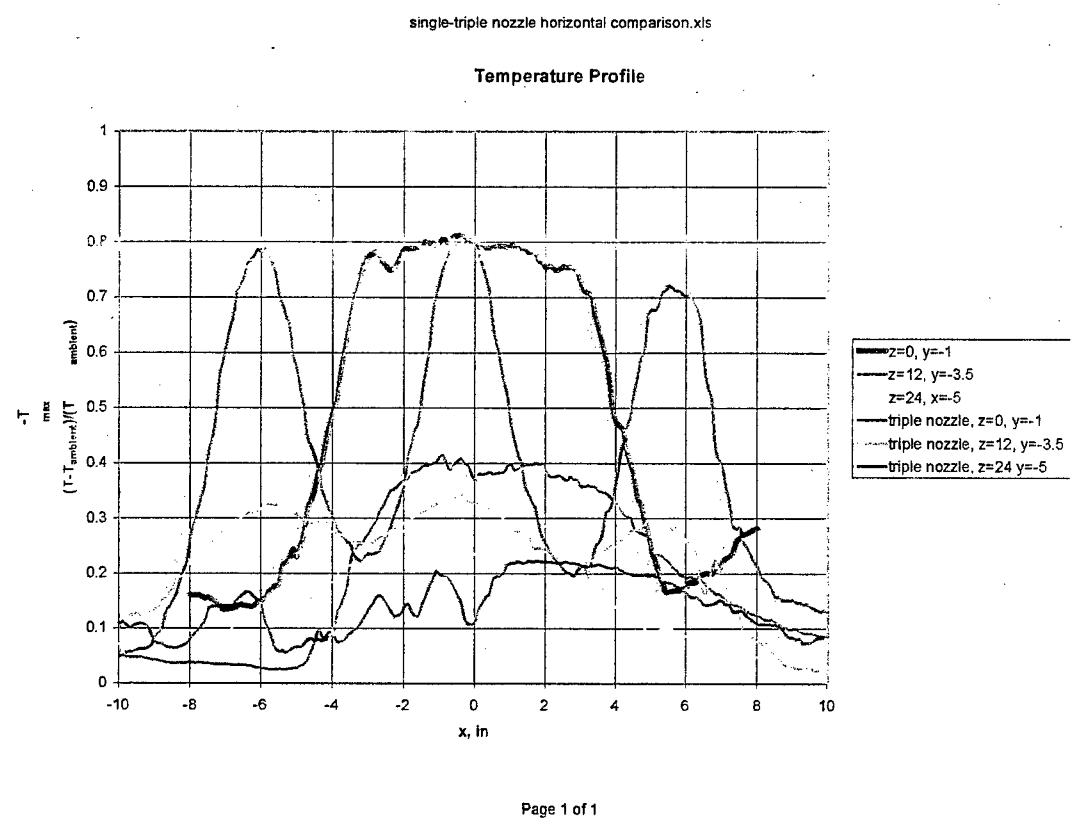


• **Figure 6: Temperature Variation with Distance from the Exhaust for the Single Nozzle**



• **Figure 7: Velocity Profile Variation with Distance from the Exhaust for the Single Nozzle**

The triple nozzle accelerated the dissipation process and created hot streaks that reduce the effective infrared signature of the exhaust by introducing cool air between the hot streaks. This must be evaluated for specific infrared seekers to determine the impact of the cool air on the size of the seeker pixels that average the radiance.



• **Figure 8: Comparison of Triple Nozzle with Triple Nozzle Temperature Profiles**

An examination of the data revealed that the thermocouple on the survey probe responded very quickly to temperature changes while immersed in the hot exhaust flow, but required several seconds to reach ambient temperature when removed from the hot flow. All temperature profiles indicated a region of warm air above the plume. This is an artifact of the time constant of the thermocouple probe. This can be corrected easily in Phase II by adjusting procedures to either stop at each data point or let the thermocouple settle or by taking data from either side of the plume.

Acknowledgements

This SBIR is under the direction of Wally Mick, TACOM under contract DAAE07-02-C-L009. The cooperation and coordination with ThermoAnalytics and Teresa Gonda has been invaluable to structuring a successful exhaust signature design model. Dr. Chris Rumsey, Dr. Robert T. Biedron, and Jeffery A. White at NASA Langley Research Center have contributed to the selection of the CFD program. Dr. T. C. Tai at the Naval Surface Warfare Center has contributed to the use of CFL3D.